Eng. & Technology, Vol.25, No.1, 2007

Study of 2-D Laminar Flow In A pipe With A sudden Contraction of Cross Sectional Area

Khudheyer S. Mushate*
Received on: 8/3/2004
Accepted on: 21/12/2004

Abstract

The paper presents a detailed numerical study of laminar flow in a pipe with sudden contraction in its cross sectional area. A computer program has been developed to analyze the laminar flow field in the pipe-sudden contraction configuration. The Navier-Stockes and energy equations are solved using primitive variables with a finite -volume (FV) solution method. The study was carried out for a Reynolds numbers range up to 750 and for different contraction ratios (d/D). The results demonstrate that the separation regions increase with the increase of Reynolds numbers. And the contraction ratio (d/D) has significant effect on these regions.

دراسة الجريان الطباقي الثنائي الأبعاد في الأنبوب ذو التضييقة المفاجئة

الخلاصة

قدم هذا البحث دراسة مفصلة للجريان الطباقي الثنائي الأبعـاد فـي الأنبـوب ذو التضـييقة المفاجئة. اذ تم تطوير برنامج حاسوب لتحليل الجريان في هذا الأنبوب. وللتنبـؤ بتصـرفات الجريان تم حل معادلات نافير ستوكس ومعادلة الطاقة عدديا باستخدام المتغيرات الأولية مـع تقنية الحجم المحدد . أجريت الدراسـة لأرقـام رينولـدز وصـلت إلـي 750 والـى نسـب تخصر(تضييقة) متعددة. بينت النتائج المستحصلة من هذه الدراسة أن مناطق الأنفصال تـزداد مع زيادة أرقام رينولدز كما أوضحت الدراسة أيضا أن لنسب التخصر تـأثير ملحـوظ علـى الجريان.

Nomenclature

d small diameter of the pipe
e contraction ratio (d/D)
L1 length of separation region upstream of the contraction.
D large diameter of the pipe
Pr Prandtl number
Re Reynolds number $\left(\frac{U_o.D}{n}\right)$
u, v velocity components
U,V dimensionless velocity components
U_o inlet velocity
X,R dimensionless coordinates
Sφ source term

* Dept. of Mechanical Engineering, UOT., Baghdad-IRAQ.

https://doi.org/10.30684/etj.25.1.2
2412-0758/University of Technology-Iraq, Baghdad, Iraq

Greek

θ dimensionless temperature

ν kinematic viscosity

Γ diffusion coefficient

Introduction

The flow through a pipe with sudden changes in cross sectional area can be found in many industrial applications. This flow is characterized by increased pressure losses due to the separation close the change in cross sectional area. As a result to these variations in pressure loss, the erosions rates, heat and mass transfer rates are increased in the regions where separated flows occurs. The flow through a pipe with sudden contraction has been studied numerically and experimentally (1-5).The researchers emphasis on the separation region before and after the contraction region. Also they interested in studying the pressure loss for the problem of interest. In this paper the problem of a flow through a pipe with sudden contraction in its cross sectional area (as shown in fig.1) is studied numerically for different Reynolds numbers (300, 500,750) and for different contraction ratios (0.5, 0.75, 0.85). In this work a finite difference procedure based on a staggered grid using control volume method is used to discretise the flow equations. SIMPLE algorithm (7) is used to predict the flow under consideration.

Theory:

The governing equations to be considered are continuity, momentum and energy equations. Constant properties are assumed. The flow was considered to be axisymetric and steady.

$$\frac{\partial u}{\partial x}+\frac{1}{r}\left(r\frac{\partial v}{\partial r}\right)=0 \qquad (1)$$

$$\frac{\partial}{x}\left(r\,u^{2}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(rruv\right)= -\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(m\frac{\partial u}{\partial x}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(mr\frac{\partial u}{\partial r}\right) \qquad (2)$$

$$\frac{\partial}{x}\left(r\,uv\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(rr\,v^{2}\right)= -\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left(m\frac{\partial v}{\partial x}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(mr\frac{\partial v}{\partial r}\right) \qquad (3)$$

$$r\,c_{p}\left(u\frac{\partial t}{\partial x}+v\frac{\partial t}{\partial y}\right)= \frac{\partial}{\partial x}\left(k\frac{\partial t}{\partial x}\right)+\frac{1}{r}\frac{\partial}{\partial}\left(kr\frac{\partial t}{\partial r}\right) \qquad (4)$$

the above equations were cast in dimensionless using D, U_o (where U_o is the average velocity at inlet), ρU^2 and T_h-T_c to scale the lengths, velocity, pressure and temperature respectively.

$$\frac{\partial U}{\partial X}+\frac{1}{R}\left(R\frac{\partial V}{\partial R}\right)=0 \qquad (5)$$

$$\frac{\partial}{\partial X}\left(U^{2}\right)+\frac{\partial}{R}\left(RUV\right)= -\frac{\partial P}{\partial X}+\frac{1}{Re}\frac{\partial}{\partial X}\left(\frac{\partial U}{\partial X}\right)+ \frac{1}{R\,Re}\frac{\partial}{\partial R}\left(R\frac{\partial U}{\partial R}\right) \qquad (6)$$

$$\frac{\partial}{\partial X}(UV)+\frac{1}{R}\frac{\partial}{R}\left(RV^2\right)=$$
$$-\frac{\partial P}{\partial R}+\frac{1}{Re}\frac{\partial}{\partial X}\left(\frac{\partial V}{\partial X}\right)+ \qquad (7)$$
$$\frac{1}{Re\,R}\frac{\partial}{\partial R}\left(R\frac{\partial V}{\partial R}\right)$$

$$U\frac{\partial q}{\partial X}+V\frac{\partial q}{\partial R}=$$
$$\frac{1}{Re\,Pr}\frac{\partial}{\partial X}\left(\frac{\partial q}{\partial X}\right)+ \qquad (8)$$
$$\frac{1}{R\,Re\,Pr}\frac{\partial}{\partial R}\left(R\frac{\partial q}{\partial R}\right)$$

Boundary conditions:

At inlet, $u = U_o$, hot fluid (θ =1)

At the walls, U = V = 0, $\frac{\partial q}{\partial R}=0$

$\frac{\partial P}{\partial n}=0$ where n vector normal to the wall

symmetry axis, $\frac{\partial U}{\partial R}=0$, V = 0.

At the exit of the pipe, $\frac{\partial U}{\partial X}=\frac{\partial V}{\partial X}=\frac{\partial q}{\partial X}=0.$

Numerical solution:

The flow equations are discredited using a finite -volume method which may be described as follows:

In the first step, the solution domain is subdivided into a finite number of control volumes (CV) by an orthogonal uniform grid. The computational points (storage locations for variables) are then located in the center of the control volumes. As seen in fig.2, all scalar variables (such as pressure and temperature are defined at the nodal points where the velocity components are defined on staggered grids centered around cell faces. Let φ be any dependent variables for which the conservation equations are as follows

$$\frac{\partial}{\partial X}(U\phi)+\frac{\partial}{\partial Y}(V\phi)=$$
$$\frac{\partial}{\partial X}\left(\Gamma\frac{\partial \phi}{\partial X}\right)+\frac{\partial}{\partial Y}\left(\Gamma\frac{\partial \phi}{\partial Y}\right)+S\phi \qquad (9)$$

where Sφ is the source term which has different expressions for different flow equations. After applying central difference for diffusion terms and upwind difference for convective terms, the above equations are discretized (reduced to algebraic equations).The discretization equations connected the values of φ for a group of grid points, usually the grids of the control volumes. The discretized form may be expressed as:

$A_P U_P = A_W U_W + A_E U_E + A_S U_S + A_N U_N + S\varphi$ (10)

where

$A_P = A_W + A_E + A_S + A_N$

Solution Methodology:

A developed computer program was used to solve the momentum discretized equations through an iterative procedure based on an integral control volume analysis with upwind finite difference and staggered grids. This procedure uses the SIMPLE algorithm developed by Patankar and Versteeg [6,7]. This procedure starts from an initial guess for all field values(typically zero). After the solution of the pressure correction equation, the nodal velocities and pressure are updated:

$U = U^{*}+U'$; $V = V^{*}+V'$; $P = P^{*}+ \alpha P'$.

Under relaxation factors were used to the components of the velocities, the pressure and heat to prevent divergence due to non- linearity in the Navier-Stokes equations.

Results and discussion

In the following, the numerical results are presented concerning the characteristics of the problem under consideration:

Fig.(3) exhibited the computational flow field with different contraction ratios and for a specified Reynolds number. It is evident that the contraction ratio has significant effect on flow field where at high contraction ratio the separation regions are increased. Also the regions down stream of the contraction are more influenced with this parameter. The figure shows that with increasing the contraction ratio the separation regions down stream of the contraction are increased and the boundary layer is developed quickly.

Fig.(4) shows the obtain pattern of the flow field for different Reynolds numbers and for contraction ratio equal to 0.2. It can be seen that the separation regions are increased with increasing the Reynolds numbers. This fact is confirmed in fig. (5) where the stream lines are increased in number and become closer to each other. Also the boundary layer growth is faster at high Reynolds numbers.

Fig.(6) shows the isotherm contours for different Reynolds numbers .It is evident that the flow has higher values of temperature at high Reynolds numbers due to friction between the layers. The isotherm contours are more thick in the separation regions especially after the contraction, and the low temperatures are shown in this regions. This may be to the pressure losses in this regions which associated with heat losses.

In Fig.(7) the reliability of computational results is checked with experimental results taken from ref (4). The comparison indicated good agreement between the numerical and experimental results.

Conclusions

The following conclusions can be drawn from this study:

- The separation regions are increased with increasing Reynolds numbers.
- The influence of separation region down stream of the contraction with increasing the contraction ratio is higher than the separation region upstream of the contraction.
- The boundary layer growth down stream of the contraction is faster at high Reynolds numbers.

References:

1. Durust F., Loy T., “Investigations of laminar flow in a pipe with sudden contration of cross sectional area”, Computer and Fluid vol. 13, 1985.

2. Cavalho M.Sc., SouZa Mendes P.R., “Heat transfer in non-newtonian ax symmetricflow in the neighborhood of a sudden contraction”, J. of Heat transf, August 1992, 114/547.

3. Vrentas J.S and Duda J. L., “Flow of Newtonian fluid through a sudden contraction”, Appl. Sc. Res. (1973).

4. Baughn J.W., Hoffman M.A., Launder, B.E., Leed. and Yap, C., “Heat transfer, temperature, and velocity measurements downstream of an abrupt expansion in a circular tube at a uniform wall temperature”,

ASME Journal of Heat transfer, vol. III, November 1989.
5. Vradis G. C., Otugen M. V., Kim S. W., Kim D. B., "Round Incompressible Jets with Initially Asymmetric Velocity Distribution", AIAA Journal, Vol. 31, No.5, May 1993, pp. 814-816.
6. Versteeg H.K., Malasekera W., "An Introduction to Computational Fluid Mechanics", 1995.
6. Palanker S. V., "Numerical Heat transfer and fluid Flow", 1980

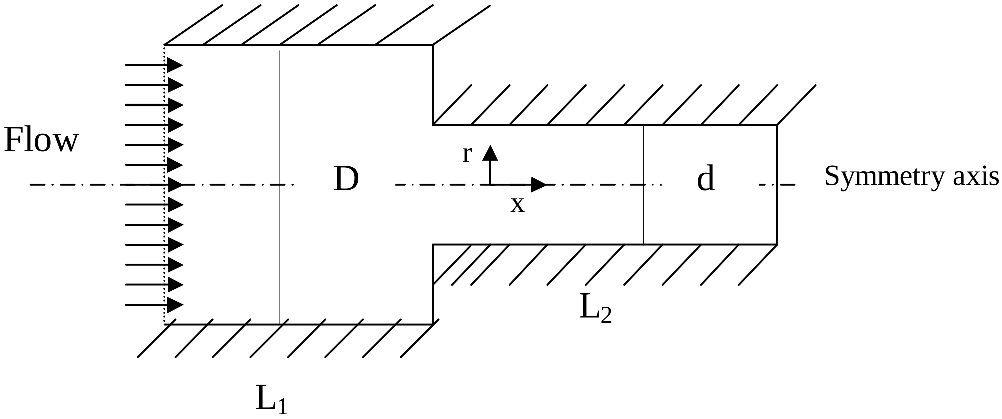


Fig. (1) configuration of the problem

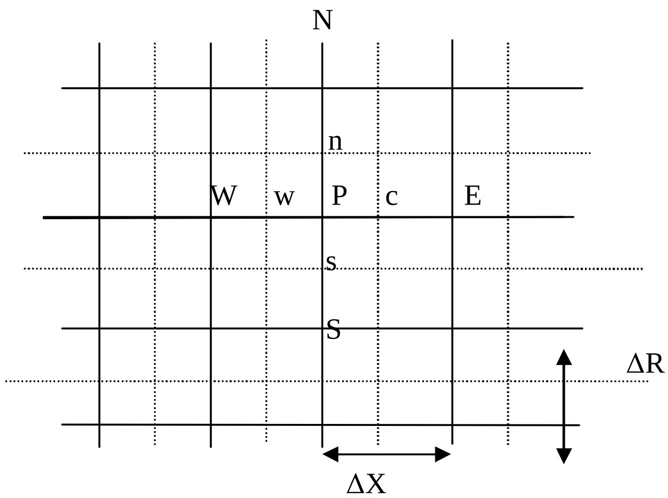


Fig.(2) two dimensional computational cell

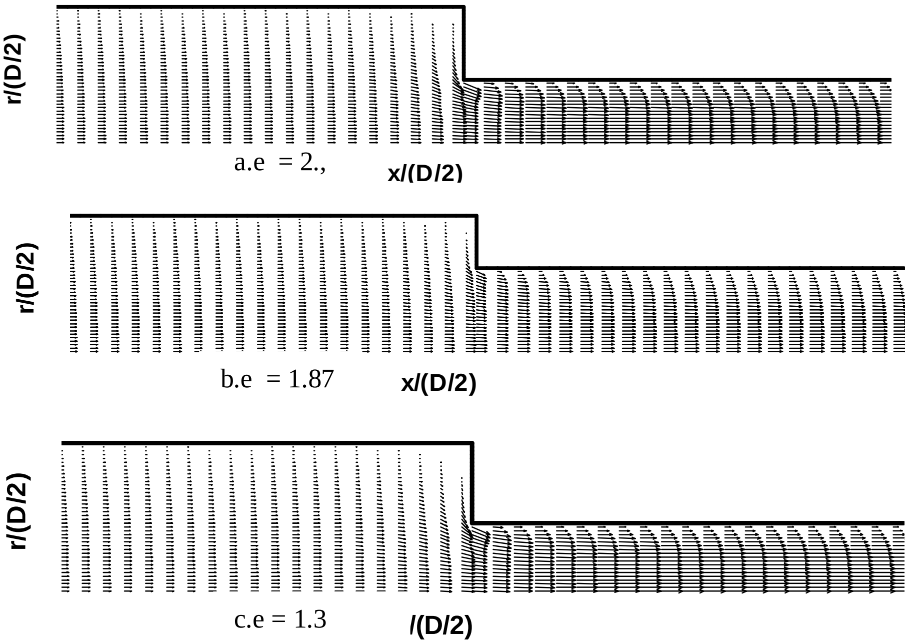


Fig. (3) computational flow field (U&V) for different contraction ratios (e), Re =500 and x/(D/2) = 16.

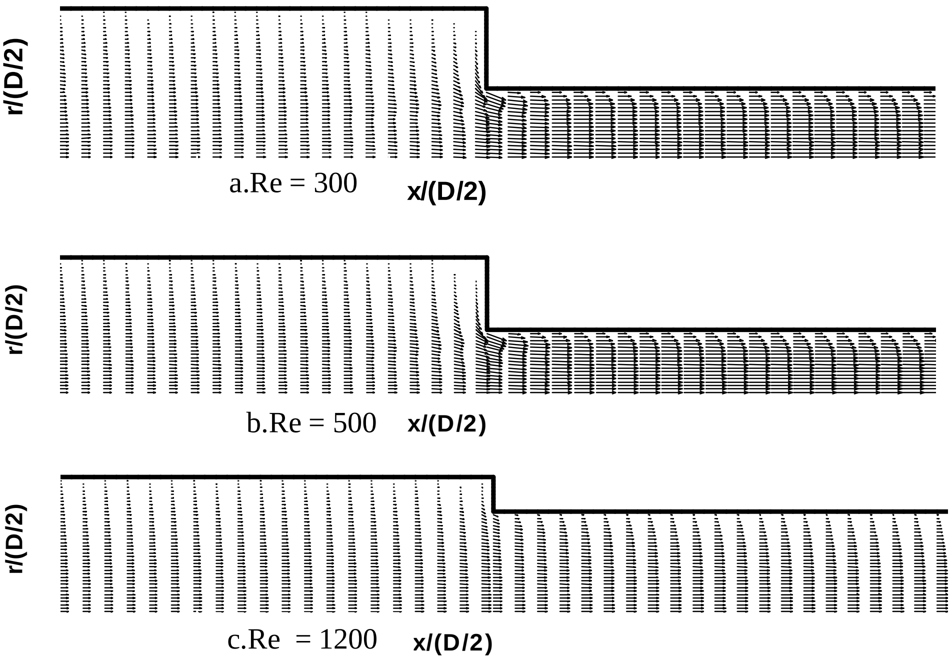
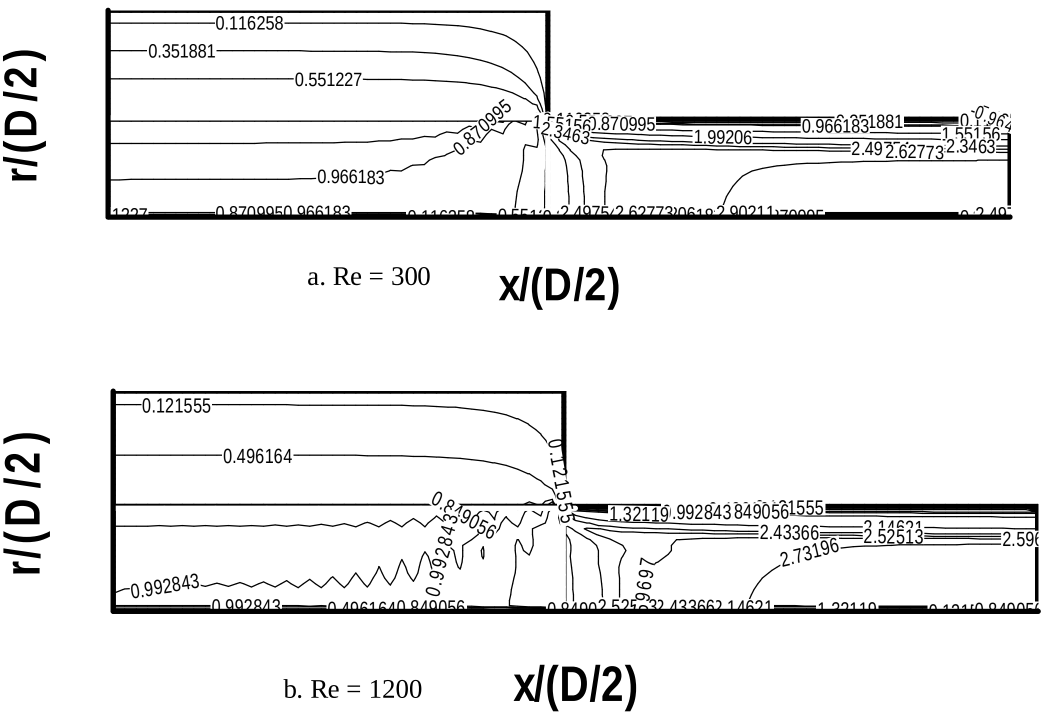


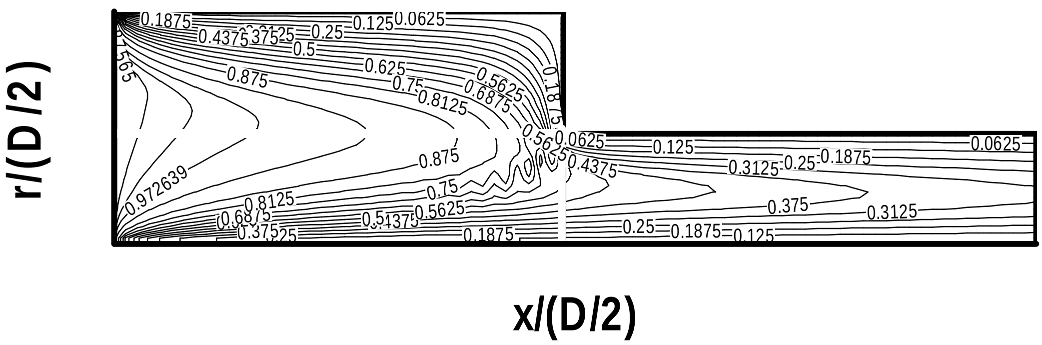
Fig. (4) computational flow field (U&V) for different Reynolds numbers and (e=2).



a. Re = 300

b. Re = 1200

Fig.(5) Contours of axial velocity for different Reynolds numbers (e = 5)



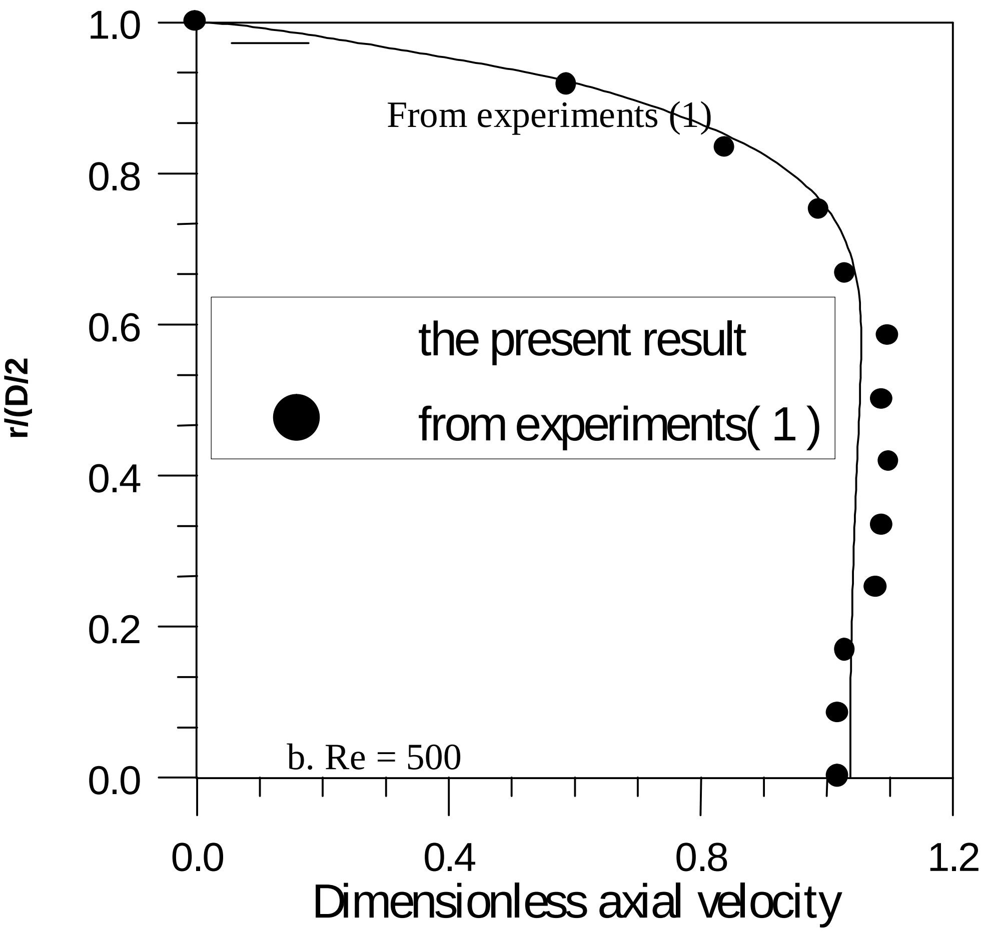


Fig.(6). Isotherm contours for different Reynolds numbers and e= 0.5

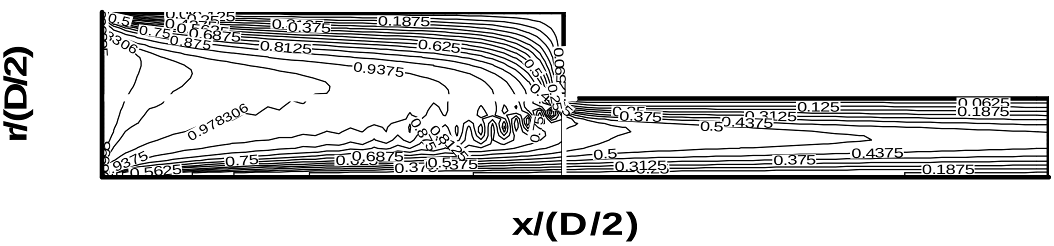


Fig.(7) comparison between numerical and experimental results